

Computational Fluid Dynamics Analysis of Cyclone Separator

Dr.I.Satyanarayana

Mechanical Engineering Department / Sri Indu Institute of Engineering and Technology, Telangana State
Affiliated to JNTUH,Hyderabad / India
ismechprofessor@gmail.com

Abstract— The paper presents a Computational Fluid Dynamics (CFD) calculation to predict and to evaluate the swirling flow gas cyclones. This study was concerned with the most common reverse flow type of cyclones where the flow enters the cyclone through a tangential inlet and leaves via an axial outlet pipe at the top of the cyclone. The Reynolds Stress Model (RSM) simulation was performed using the commercial package FLUENT6.3.26. In the present investigation the pressures and velocities distribution has been generated using CFD. The problem of modeling highly swirling flow is overcome by means of an algebraic turbulence model, and all the features of the experimentally observed gas flow in a cyclone are shown to be computationally reproduced. It was found that CFD simulations predict excellently the axial and tangential velocity with an average deviation of ± 0.5 m/s from the presented experimental data. The physical mechanism for prolonged cyclone cylindrical body and vortex finder has also been successfully elucidated. Specifically, results obtained from the computer modeling exercise have demonstrated that CFD with RSM turbulence model is suitable for modeling a flow field and hydrodynamic of cyclone.

Keywords— *Keywords-- computational fluid dynamics (CFD), Reynolds Stress, Model, pressure and velocity distribution*

I. INTRODUCTION

Cyclone separators are used to remove particulate from a fluid stream. This is accomplished by Centrifugal forces that force the particles to dislodge from the fluid flow. The fluid exits through the vortex finder at the top of the cyclone while, due to gravitational forces, the particles fall to the bottom of the cyclone for collection. Pressure loss and collection efficiency are the two most important aspects of a cyclone, because both directly affect the process costs. Cyclone separators are used to remove particulate from a fluid stream. This is accomplished by centrifugal forces that force the partic Experimental and numerical studies have been carried out in the last few decades to develop a better understanding of the flow field inside the cyclone separators. In the early years, empirical models were built (e.g. Shepherd & Lapple, 1939; Lapple, 1951; Barth, 1956; Tengbergen, 1965; Sproul, 1970; Leith & Licht, 1972; Blachman & Lippmann, 1974; Dietz, 1981 and Saltzmann, 1984) to predict the performance of industrial cyclones. However, these models were built based on the data from much smaller sampling cyclones therefore; they could not achieve desired efficiency on industrial scales as the industrial cyclone operates in the turbulent regime while sampling cyclones operate under the transitional conditions. One of the major drawbacks of these empirical models is the fact that they ignore two critical factors that determine the performance of a cyclone namely the unsteadiness and asymmetry. Many flow phenomena such as high turbulence, flow reversal, forces, the particles fall to the bottom of the cyclone for collection. Pressure loss and collection efficiency are the two most important aspects of a cyclone, because both directly affect the process costs.

Computational fluid dynamics (CFD) models on the other hand can accurately capture these aspects and thus can take a significant role in analyzing the hydrodynamics of cyclone separators. A validated CFD model can be a valuable tool in developing optimal design for a given set of operating conditions. However, cyclone separators pose a peculiar fluid flow problem. The flow in cyclone separators is characterized by an inherently unsteady, highly anisotropic turbulent field in a confined, strongly swirling flow. A successful simulation requires proper resolution of these flow features. Time dependent turbulence approaches such as large eddy simulation (LES) or direct numerical simulation (DNS) should be used for such flows. However, these techniques are computationally intensive and although possible, are not practical for many industrial applications. Several attempts have been made to overcome this drawback. Turbulence models based on higher-order closure, like the Reynolds Stress Model, RSM, along with unsteady Reynolds averaged Navier – Stokes (RANS) formulation have shown reasonable prediction capabilities (Jakirlic & Hanjalic, 2002). The presence of solids poses additional complexity and multiphase models need to be used to resolve the flow of both the phases.

Operation principle of cyclone dust separators consists in employment of centrifugal force cause by vortex movement of liquids. The multi- phase blend of liquid and solids is supplied to the upper part of a cyclone. Vortex low of the blend through the

cyclone leads to concentration of the solid phase nearby walls of the outermost cylinder. Since the descending spiral shape of the low channel the deposited solids are transferred downwards to the discharge port. In turn, the liquid phase is reversed and then transferred upward where it is released outside the unit through outlet channel aligned with the central axis of the unit. In this review the CFD simulations for cyclone separators. Important cyclone characteristics such as the collection efficiency, pressure and velocity fields have been discussed and compared with the experimental data. Several significant parameters such as the effect of geometrical designs, inlet velocity, particle diameter and particle loading, high temperature and pressure have also been analyzed. These calculations of the continuous phase flow were the basis for modeling the behavior of the solid particles in the cyclone separator. Particle trajectories, velocities, pressures and the pressure drops have been studied in the present thesis. In the present work the pressures and velocities distribution have been generated using CFD. The pressure drops have been evaluated for the existing design and the modified design. Significant pressure drops have been observed in the optimized model.

II. LITERATURE SURVEY

CFD helps to understand the complex flow patterns in cyclones and develop efficient cyclone with low pressure drops and high separation efficiency. Cyclones used in many technical applications [1], such as physical separation processes (especially dust from a gas stream) and chemical reactions (combustion of solid low calorific value fuels). In spite of the wide use and simple geometry, the real working principles of a cyclone chamber are far from being fully understood, mainly owing to the extreme complexity of the swirling turbulent flow field inside the device [2]. The gas stream enters the cyclone tangentially and force by its geometry into a vortex motion in the cylindrical section, spiraling downwards until the conical section is reached. In this part of the device, the centrifugal forces can be several times greater than gravity, contributing to particle separation. Then, the gas flow is deflected upwards and outflows from the exhaust duct (also called vortex finder), maintaining the swirling motion. The complexity of the flow pattern inside the chamber is due to the high turbulence level, strong anisotropy, three-dimensionality (3-D) and possible non stationary features typical of highly swirling motions, so that both experimental analysis and numerical simulations become notably difficult. Probably, this is the cause that basic cyclone design has evolved very little from the first applications.

Analytical solutions for the gas motion have been reported in the past [3, 4], but, due to the approximations introduced, their results are only valid in delineating the main features of the vortex flow. Numerical solutions have been also developed [5, 6], but they are strongly dependent on the turbulence model used and satisfactory agreement with experimental data has been achieved using the second-order Reynolds-stress model (RSM) as closure assumption for the turbulence modelling in a general purpose finite volume code [7, 8, 9]. Early experimental investigators of the cyclone chamber behaviour used Pitot tubes [10] and hot-wire anemometry [11]. They aimed first at determining the most efficient shape of a cyclone and then at understanding the turbulent flow structure, but the two techniques have several drawbacks and the reliability of their results cannot be guaranteed. Detailed measurements of the gas flow in a cyclone became possible only recently with the availability of Laser Doppler Velocimetry (LDV) and revealed a very complex structure with 3-D time dependent instabilities originated by the strongly swirling flow [12].

III. CYCLONE FLOW AND PRESSURE FIELDS

The collection efficiency and pressure drop performance of the cyclone separator are a direct result of the flow patterns of gas and solid and pressure field inside the cyclone. In a time averaged basis, the dominant flow feature in a cyclone separator is a vortical flow that can be described as the Rankine vortex, which is a combination of a free outer vortex and a forced inner vortex. Apart from the inlet gas velocity and geometrical parameters, the wall friction and solid loading also influence the strength of the vortex. The empirical models often neglect the later two aspects and hence are limited in their application. Computational modelling is needed to resolve the velocity and pressure fields (Kim et al., 1990, Hoekstra et al., 1999, Ma et al., 2000, Slack et al. 2000 and Solero et al 2002).

A. Axial velocity

The axial velocity of the gas phase is a major influence in the transportation of particles to the collection device. Empirical models based on the double vortex structure postulate radially constant values for the downward flow in the outer vortex and upward flow in the inner vortex. Both these values are zero at the axial position where the vortex ends. In reality, the profiles are

not flat but exhibit maxima and minima. Typically the downward flow shows a maximum near the walls, while the upward flow shows either a maximum or a minimum at the symmetry axis. The diameter of the swirl of gas entering the vortex finder is larger than the vortex finder diameter itself. Consequently, the gas velocity expected to increase and peak at the vortex finder either on the centre or at the sides. This results in an inverted V or an inverted W shaped profile as seen in figures 4a and 4b for the inner vortex. The V pattern forms an axial velocity maximum at the vortex core of the cyclone while the W pattern forms an axial velocity maximum at the vortex finder radius with a minimum at the vortex core.

B. Radial velocity

The radial velocity affects the particle bypass and is an important factor in analyzing the particle collection and losses of efficiency. Frequently the radial velocity is assumed to be of much lesser magnitude than the other components. However, this is valid only in the outer vortex, and especially near the vortex finder, the radial velocity increases rapidly towards the vortex core (Muschelknautz, 1972).

C. Tangential velocity

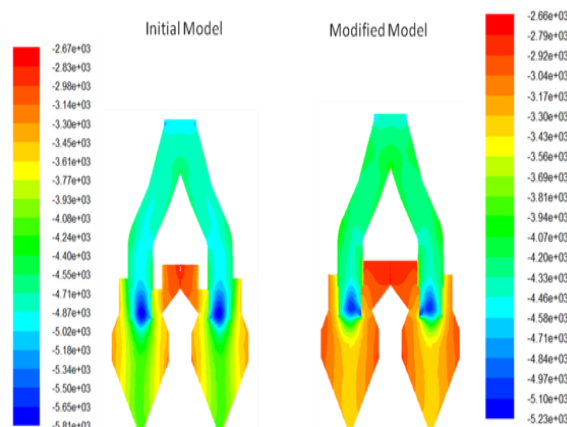
The flow within a cyclone is dominated by the tangential velocity and strong shear in the radial direction which results in a centrifugal force that determines the particle separation. Subsequently, much discussion within cyclone separator studies is focused on the tangential velocity (Cortes & Gil, 2007).

IV. DESIGN CONSIDERATIONS

A small inlet and outlet therefore result in the separation of smaller particles. The depth and diameter of the body should be as large as possible because the former determines the radial component of the gas velocity and later controls the tangential component at any radius. In general the larger the particles, the larger should be the separator diameter, because greater is the radius at which they rotate, the greater too is the inlet velocity which can be used without causing turbulence within the separator. The factor which ultimately settles the maximum size is, the cost. Because the separating power is directly related to the throughput of gas, the cyclone separator is not very flexible though its efficiency can be improved at low throughputs by restricting the area of the inlet with a damper and thereby increasing the velocity. However it is better to use a number of cyclones in parallel and to keep the load on each approximately the same. The effect of the arrangement and size of the gas inlet and outlet has been investigated and it has been found that the inlet angle should be of the order of 180°. The efficiency of the cyclone separator is greater for large than for small particles and it increases with the throughput until the point is reached where excessive turbulence is created. The efficiency of collection plotted against particle size for an experiment separator for which the theoretical “cut” occurs at about 10µm. It may be noted that an appreciable quantity of fine material is collected, largely as a result of agglomeration, and that some of the coarse material is lost with the result that a sharp cut is not obtained. A number of formulations have been developed for determining the fractional cyclone efficiency η for a given size particle. Fractional efficiency is defined as the fraction of particles of a given size collected in the cyclone, compared to those of that size going into the cyclone.

$$\text{Collection efficiency} = \frac{[\text{inlet loading} - \text{outlet loading}]}{[\text{inlet loading}]} \times 100.$$

Various commercial packages are used for modeling of cyclone separator .Pro-E is used for the modeling and meshing is done in HYPERMESH which is used as preprocessor. ANSYS FLUENT SOLVER is used for the post processor.



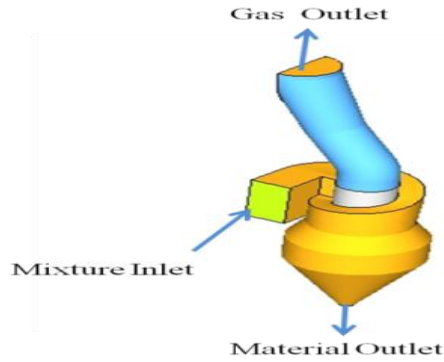


Fig.1 Model generated in CFD

V. RESULTS AND DISCUSSIONS

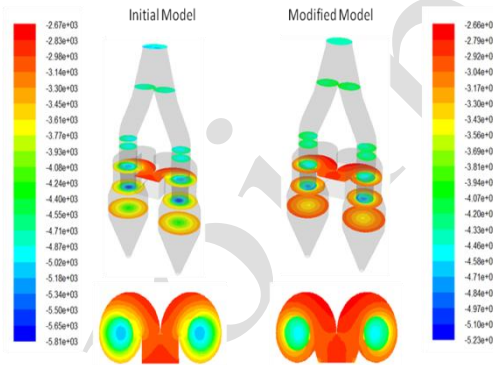


Fig 2 Pressure Contours of mixture at top plane of cyclone (Pa)

From figures Pressure drop in the initial model observed is 2350 Pa , Pressure drop in the Modified model observed is 1790 Pa and Pressure drop is reduced to 560 Pa in the Modified model.

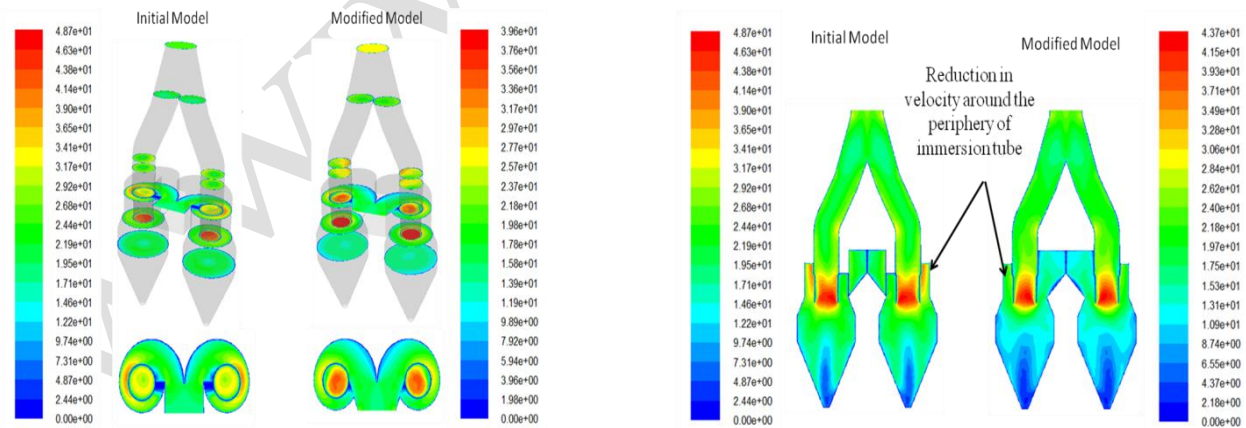


Fig . 4 Velocity Contours of mixture at mid plane of cyclone assembly (m/s)

Fig . 3 Velocity Contours of gas at Top plane of cyclone (m/s)

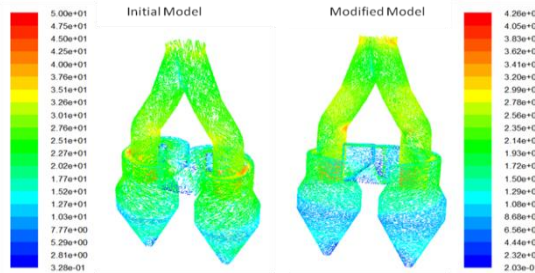


Fig . 5 Velocity Vectors of gas on outer surface of cyclone (m/s)

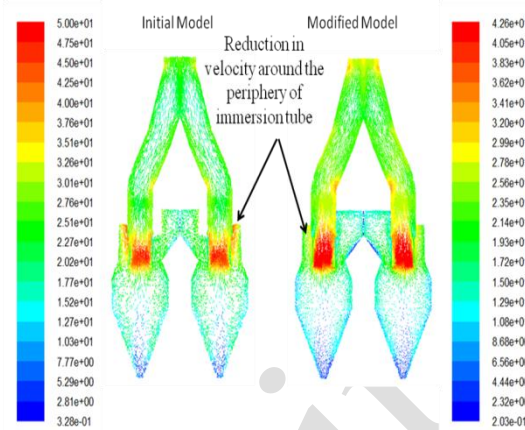


Fig.6 Path lines colored by Velocity (m/s)

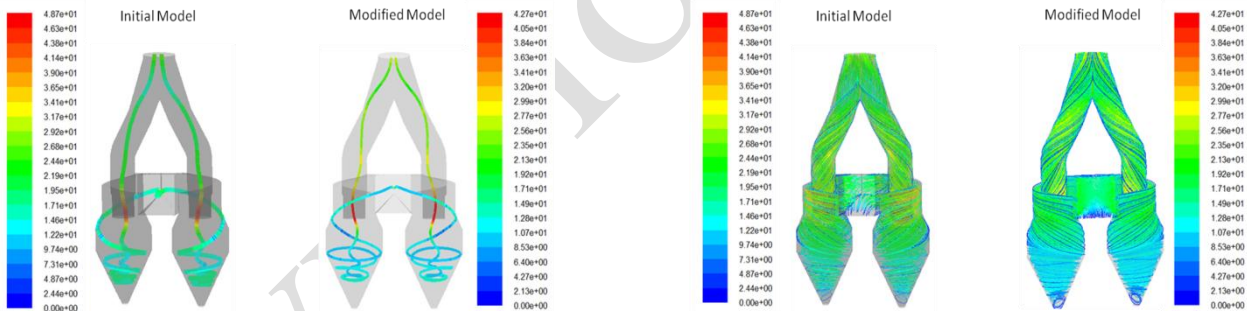


Fig.7 Path lines colored by Velocity (m/s)

VI. CONCLUSIONS

In order to get the more efficiency of cyclone separator we need to reduce the pressure drop across intake and exit. So I have modified the initial model inlet and maintained some angularity through which cement particles and gaseous mixtures are entering into the cyclone separator. Pressure drop from inlet to outlet in the initial model observed is 2350 Pa. Pressure drop from inlet to outlet in the Modified model observed is 1790 Pa. Pressure drop is reduced to 560 Pa in the Modified model. Hence the modified model with little modification at the inlet angle can be proposed for the industrial applications. The proposed model provides a convenient way to study the effects of variables related to operational conditions, cyclone geometry and particle properties plays an important role to optimize the design and control of cyclone process.

The geometrical parameters in cyclone separators affect significantly the flow field and performance parameters. There are seven geometrical parameters, which can be classified into four classes, namely, the inlet dimensions (height and width), the

vortex finder dimensions (diameter and length), the cyclone height (cone and barrel) and the cone-tip diameter. These dimensions do not have the same influence on the cyclone performance. Consequently, the first step was to identify the most significant parameters and the possible interaction between them.

Due to their simple and robust construction, cyclone separators are widely used in the chemical and process industries. In spite of their simple construction, flow patterns inside cyclones are highly complex and not fully understood. Understanding the flow is critical in accessing their performance and CFD is a useful tool in providing this information. However, due to the very nature of the flow, the application of CFD should be exercised with prudence.

TABLE 1 MATERIAL PROPERTIES

SI No	Gas Properties
1	Density = 1.293 kg/m ³
2	Thermal Conductivity = 0.0242 W/m-K
3	Cp = 1006.43 J/kg-K
4	Dynamic Viscosity = 1.7894 e-5 kg/m-sec
SI No	Cement Particles Properties
1	Density = 1502 kg/m ³
2	Thermal Conductivity = 2 W/m-K
3	Cp = 920 J/kg-K
4	Dynamic Viscosity = 0.001 kg/m-sec

TABLE 2 BOUNDARY CONDITIONS

Inlet Boundary Conditions	
Pressure (Pa)	-2668.752368
Volume fraction of 45microns material	0.000321841
Temp at Inlet (K)	365

Gas Outlet Boundary Conditions	
Velocity (m/s)	-26.63853941
Volume fraction of 45microns material	1.61E-05
Temp at Outlet (K)	365

Material (45 microns) Outlet Boundary Conditions	
Velocity (m/s)	-0.116455188
Volume fraction of 45microns material	1
Temp at Outlet (K)	365

SI No	Process parameters considered in CFD Analysis
1	Cyclone efficiency = 95%
2	Cement Particle size = 45 microns
3	Particulate mass flow rate = 64 TPH

References

- [1] A. K. Gupta, D. G. Lilley and N. Syred, *Swirl Flows*, Abacus Press, Tunbridge Wells, 1984.
- [2] G. Solero and A. Coghe, Experimental fluid dynamic characterisation of a cyclone chamber, *Experimental Thermal and Fluid Science*, 27 (2002) pp. 87-96.
- [3] C. C. Hwang, H. Q. Shen, G. Zhu and M. M. Khonsary, On the main flow pattern in hydrocyclones, *J. Fluids Eng.*, 115 (1983) pp. 21–25.
- [4] L. X. Zhou and S. L. Soo, Gas–solid flow and collection of solids in a cyclone separator, *Powder Technol.*, 63 (1990) pp. 45–53.
- [5] F. Boysan, W. H. Ayers and J. Swithenbank, A fundamental mathematical modelling approach to cyclone design, *Trans. IChemE*, 60 (1982) pp. 222–230.
- [6] S. Nieh and J. Zhang, Simulation of the strongly swirling aerodynamic field in a vortex combustor, *J. Fluids Eng.*, 114 (1992) pp. 367–374.
- [7] E. D. Cristea, A. Coghe, G. Solero, P. Conti, 3-D numerical computation and validation of high solid loading flow inside a gas cyclone separator, in *Proceedings of the 1998 ASME Fluids Engineering Summer Meeting*, Washington, DC, 1998.



- [8] J. Gimbut, T. G. Chuah, A. Fakhru'l-Razi and Thomas S. Y. Choong, The influence of temperature and inlet velocity on cyclone pressure drop: a CFD study, *Chem. Eng. Process.*, 44 (2005) pp. 7-12.
- [9] J. Gimbut, Thomas S. Y. Choong, T. G. Chuah, A. Fakhru'l-Razi, A CFD study on the prediction of cyclone collection efficiency, *Int. Journal of Computational Engineering Science*, (2004) (in press).
- [10] A. J. Linden, Investigations into cyclone dust collectors, *Proc. Inst. Mech. Eng.*, 130 (1949) pp. 233–251.
- [11] B. P. Ustimenko and M.A. Bukhman, Turbulent flow structure in a cyclone chamber, *Teploenergetika*, 15 (1968) 64–67.
- [12] T. O'Doherty, R. Jarczyewski, C. J. Bates, N. Syred, Velocity characteristics of cyclone combustors, Vol. 1: Laser Anemometry, ASME, 1991.
- [13] G. E. Klinzing R.D. Marcus, F. Rizk and Leung, *Pneumatic Conveying of Solid*, 2nd ed., Chapman and Hall, new York, 1997.
- [14] S. M. Fraser, A. M. Abdel Rasek and M. Z. Abdullah, (1997) Computational and experimental investigation in a cyclone dust separator, *Proc. Instn. Mech. Engrs.*, 211 (Part E) (1997) pp. 247-257.
- [15] W. Peng, A. C. Hoffmann, P. J. A. J. Boot, A. Udding, H. W. A. Dries, A. Ekker and J. Kater, Flow pattern in reverse-flow centrifugal separators, *Powder Technol.*, 127 (2002) pp. 212-222.
- [16] A. K. Coker, Understand cyclone design, *Chem. Eng. Progr.*, 28 (1993) pp. 51–55.
- [17] Y. Zhu and K. W. Lee, Experimental study on small cyclones operating at high flow rates, *J. Aerosol Sci.*, 30 (1999) pp. 1303–1315.
- [18] J. C. Kim and K. W. Lee (1990) Experimental study of particle collection by small cyclones, *Aerosol Science and Technology*, 12 (1990) pp. 1003–1015.
- [19] M. E. Moore and A. R. Mcfarland, (1993) Performance modeling single-inlet aerosol sampling cyclone, *Environmental Science and Technology*, 27 (1993) pp. 1842–1848.